ECEN 250 Lab2- Thevenin and Norton Equivalents, and Power transfer

Name: Brodric Young

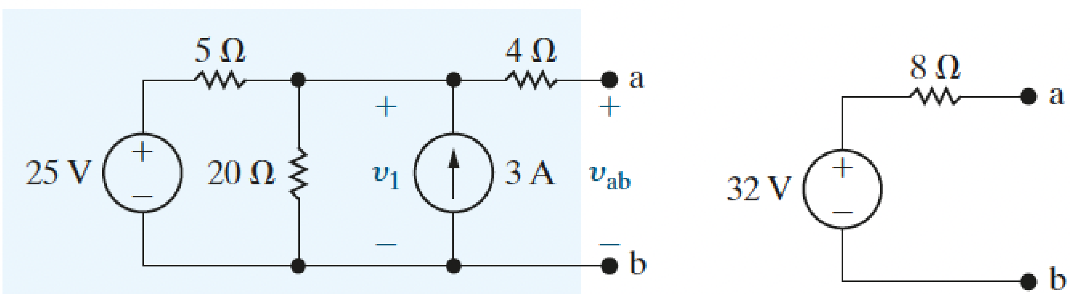
Purposes:

- Verify the Thevenin Equivalent circuit concept through simulation
- Verify the Norton Equivalent circuit concept through simulation
- Understand power transfer concepts

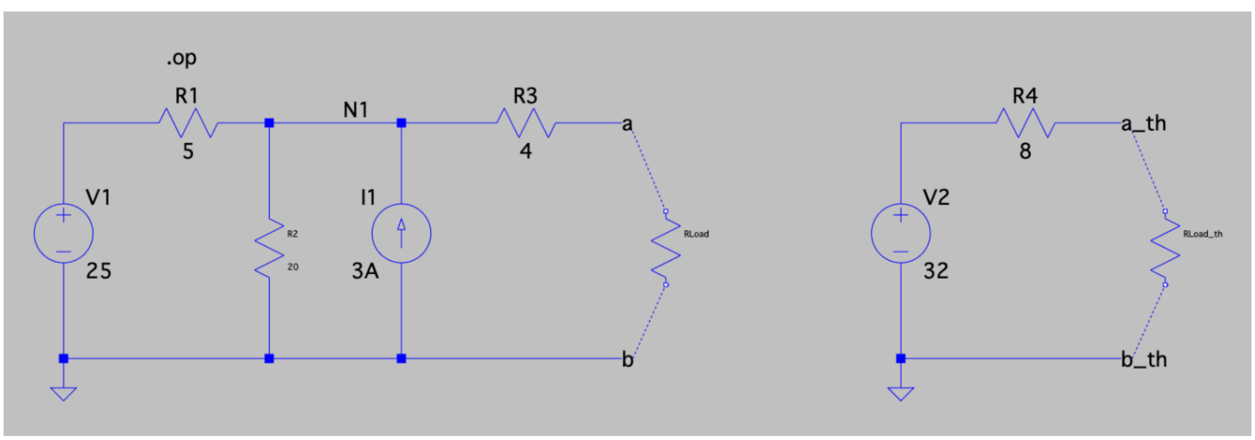
Procedure:

<u>Part 1 - SPICE simulation</u>

Simulate the following circuits and paste a screenshot of the results under each schematic:



Figures 4.47 and 4.49 in the textbook (Example 4.14)



Example 4.14 - A circuit used to illustrate a Thevenin equivalent

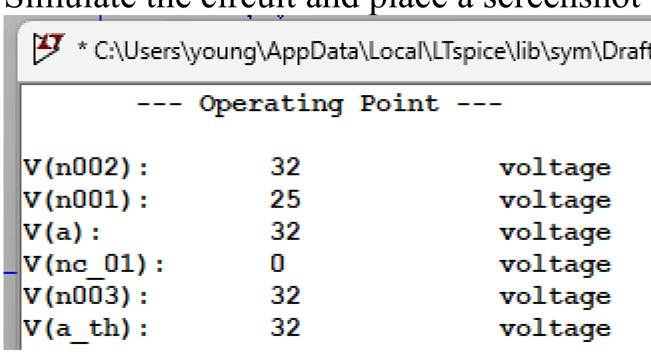
Open LTspice and create the schematic of Example 4.14.

Use the following LTspice function keys and shortcuts:

- Esc to exit
- F2 to place a component
    - "r" for resistor
    - "vo" for voltage source
    - "cu" for current source
- F3 to place a wire
- F4 to place a net name (like N1, a, and b)
- F4 to place a ground symbol
- F5 to delete
- F6 to copy
- F7 to move
- F8 to drag
- Ctrl-r to rotate
- Ctrl-e to mirror
- Right-click on text to change
- Right-click -> Draft -> SPICE directive to place a command

You will need to place the command ".op" in the schematic window to instruct LTspice to find the operating point of the circuit (other commands you might use in the future are .tran, .ac, .dc, .tf, and .noise).

Simulate the circuit and place a screenshot of the results below:


* C:\Users\young\AppData\Local\LTspice\lib\sym\Draft

```
       --- Operating Point ---

V(n002):       32            voltage
V(n001):       25            voltage
V(a):          32            voltage
V(nc_01):      0             voltage
V(n003):       32            voltage
V(a_th):       32            voltage
```

- Compare "a" with "a_th" recording your results in Table 1 to "no load"

- Add a load resistor to complete Va and Va_th columns of Table 1 (no need for the screenshot)

  Note: LTspice will recognize "1k" as 1000, but don't use "1M" to represent 1,000,000. It will be interpreted as 0.001. Use (1Meg) to represent 1,000,000.
- Create a circuit for the Norton equivalent circuit to complete the Va_norton column, and place a screenshot of your circuit schematic below Table1:

Table 1

Load Resistance	Va	Va_th	Va_norton (see instructions below)
no load	32V	32V	32V
8Ω	16V	16V	16V
100Ω	29.6V	29.6V	29.6V
1kΩ	31.7V	31.7V	31.7V
1MegΩ	31.9V	31.9V	31.9V

COMMANDS

SPICE Analysis

.OP	find the DC operating point
.TRAN	perform nonlinear transient analysis
.AC	perform small signal AC analysis
.DC	perform DC source sweep analysis
.TF	find the DC small-signal transfer function
.NOISE	perform noise analysis

SPICE Directives

.BACKANNO	annotate subcircuit pin names on port currents
.END	end of netlist
.ENDS	end of subcircuit definition
.FOUR	compute fourier component
.FUNC	user defined functions
.FERRET	download a file from URL
.GLOBAL	declare global nodes
.IC	set initial conditions
.INCLUDE	include file
.LIB	include library
.LOADBIAS	load a previously solved DC solution
.MACHINE	arbitrary state machine
.MEASURE	evaluate user-defined electrical quantities
.MODEL	define a SPICE model
.NET	compute network parameters in .AC analysis
.NODESET	supply hints for initial DC solution
.OPTIONS	set simulator options
.PARAM	user-defined parameters
.SAVE	limit the quantity of saved data
.SAVEBIAS	save operating point to disk
.STEP	parameter sweeps
.SUBCKT	define a subcircuit
.TEMP	temperature sweeps
.TEXT	user-defined string
.WAVE	write selected nodes to a .WAV file

ANALOG DEVICES

LTspice®

Fast • Free • Unlimited

SHORTCUTS

Schematic and Symbol Editing Modes

Windows	Choose Mode then select component Exit mode: Press [Esc] or right-click	Mac
[F5] or [Delete] or [Ctrl]X	cut/delete	[F5]
[F6] or [Ctrl]C	copy/duplicate*	[F6]
[F7]	move* *unselected wires remain*	[F7]
[F8]	drag* *connected wires adjust*	[F8]
[Esc]	exit current mode *or right-click*	[Esc]

Zoom and Grid

Windows	Zoom in and out with scroll wheel or track pad pinch	Mac
[Ctrl]Z	**Schematic** zoom area (drag over area) zoom in (click on scheme) **Waveform** zoom area is default mode [F9] for previous zoom **Symbol** zoom in	
[Ctrl]B	zoom out	
[Space]	zoom to fit (schematic viewer)	[Space]
[Ctrl]E	zoom extents (waveform viewer)	
[Ctrl]G	toggle grid	

Place Component Modes*

Windows	Press [Esc] or right-click to exit place component mode	Mac
R	resistor	R
C	capacitor	C
L	inductor	L
D	diode	D
G	ground	G
V	voltage	V
S	spice directive *right-click text field to open "Help me Edit" dialog*	S
T	text/comment	T
[F2]	component	[F2]
[F3]	draw wire	[F3]
[F4]	label net	[F4]
	bus tap	B

*Rotate and Mirror

Windows	*enabled in place modes	Mac
[Ctrl]R	rotate	# R
[Ctrl]E	mirror	# E

Undo/ Redo

Windows	### Levels of Undo	Mac
[F9]	undo	[F9] or # Z
5 [F9] or [Ctrl]5 Z	redo	5 [F9] or # 5 Z

TRICKS

Waveforms

Windows	when clicking waveform label	Mac
click	add cursor and see measure	click
[Alt] click	highlight corresponding net in schematic	# click
[Ctrl] click	integrate waveform	[Ctrl] click

Schematics

Windows		Mac
[Alt] click	**component:** plot instantaneous power **wire:** plot current	# click
hold [Ctrl]	draw wires at an angle	hold [Shift]
[Ctrl][Alt][Shift] H	show hidden component values/text, e.g. parallel or series resistance and capacitance	
	any text preceded by an underscore, e.g. "_FAULT" is displayed with an overbar, active low, signal	

NUMBERS

Prefixes (Case Insensitive)

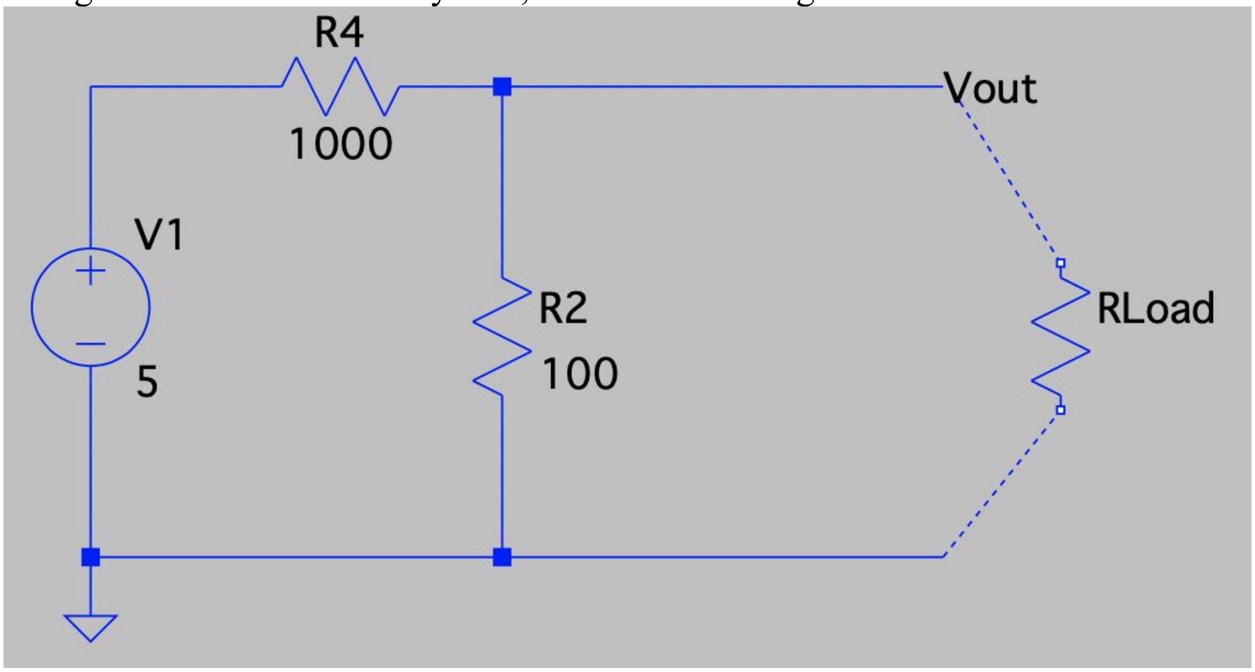
LTspice	Means	Value
T or t	tera	10^{12}
G or g	giga	10^{9}
meg	mega	10^{6}
K or k	kilo	10^{3}
M or m	milli	10^{-3}
U or u	micro	10^{-6}
N or n	nano	10^{-9}
P or p	pico	10^{-12}
F or f	femto	10^{-15}

Constants

LTspice	Means
e	Euler's number
pi	π
k	Boltzmann constant
q	charge constant
true	1
false	0
mil	25.4×10^{-6}m

Part 2 - Maximum Power

Using the class breadboard system, wire the following circuit:



Predict the load resistance that would yield the highest power delivered to the load:

Equivalent to the resistance in parallel with the load

Measure the power across the following loads:

	Load Voltage	Load Current (V_{Load}/R_{Load})	Load Power
10Ω	0.6V	6mA	0.36mW
100Ω	0.24V	2.4mA	0.58mW
10kΩ	0.46V	0.46mA	0.21mW
100kΩ	0.46V	0.046mA	0.021W

Which load yields the highest power? Does it make sense?

The 100 ohm load yielded the highest power, it does make sense because the current will be flowing equally through both 100 ohm paths and would make the highest power.

Conclusions (write a conclusion statement that discusses each of the purposes of the lab):

In this lab we verified the Thevenin and Norton equivalents circuits through simulation. We had a circuit and then created its equivalent and the measurements across the load for each was the same in each case of differing load resistances. This proves that the load "sees" the same thing whether it's a big complicated circuit or its simple Thevenin or Norton equivalent. We also learned more about power transfer concepts through building a physical circuit and experimenting with which load resistance came up with the highest power output. We had to double check the numbers we calculated a few times, but once we got it figured out everything checked out and it turned out to be equal to the resistance the load was in parallel with.